Yang, L., Carreño, . A. A., Gil, A. J., & Bonet, J. (2014). An immersed structural potential method for incompressible flexible/rigid/multi-phase flow interaction. In 22nd ACME Conference on Computational Mechanics, Exeter, UK, 2014.. (pp. 226-229). University of Exeter.

Proceedings of the 22nd UK Conference of the
Association for Computational Mechanics in Engineering
2 - 4 April 2014, University of Exeter, Exeter

AN IMMERSED STRUCTURAL POTENTIAL METHOD FRAMEWORK FOR INCOMPRESSIBLE FLEXIBLE/RIGID/MULTI-PHASE FLOW INTERACTION

L. Yang[1], A. Arranz Carreño[2], A.J. Gil[1] and J. Bonet[1]

[1]Civil & Computational Engineering Centre, College of Engineering, Swansea University, Swansea SA2 8PP
[2]Department of Mathematics, College of Science, Swansea University, Swansea SA2 8PP

{530567; a.arranz-carreno; a.j.gil; j.bonet}@swansea.ac.uk

ABSTRACT

In this paper, a general framework for the computational simulation of Fluid-Structure Interaction (FSI) problems involving rigid/flexible solids and multiphase flows is presented. The proposed methodology builds upon the Immersed Structural Potential Method (ISPM) developed by the authors [1, 2] for the simulation of single-phase FSI problems. Several numerical examples are presented to showcase and benchmark the proposed methodology in the solution of complex multi-body multi-phase problems.

Key Words: *Immersed Structural Potential Method (ISPM); rigid body constrains; level set method; augmented Lagrangian method*

1. Introduction

The numerical solution of incompressible flexible/rigid/multi-phase flow interaction problems is extremely important and commonplace in many engineering applications: from costal engineering, civil engineering to ship hydrodynamics. In general terms, two main families of methodologies have been used in practice: body-fitted approaches [3] and immersed type methods [1, 2, 4]. Methods of each family have some strengths and weaknesses, but within the body-fitted methodologies, the main disadvantage is the cost of mesh update and re-meshing algorithms, a factor particularly important in the case of three-dimensional simulations. As an alternative to such methodologies, Peskin [4] originally introduced the Immersed Boundary Method (IBM) for the solution of heart valve problems, where the computation is performed on a background Cartesian grid and a body force is added to the fluid to account for the presence of a solid.

There have been several extensions of immersed methodologies since their inception. One such extension to the original IBM is the Immersed Structural Potential Method (ISPM), introduced by the authors [1, 2] for the solution of single-phase FSI problems with highly deformable structures, such as those present in typical haemodynamic problems. In such scenarios, the methodology is robust and efficient, but the consideration of rigid, or very stiff structures, can be a limiting factor. For such problems, the authors have opted for extending the methodology by adding a Lagrange multiplier field to enforce the rigid body constraints, and solve the arising mixed formulation using a Least Squares projection approach [5]. Moreover, for the case of immersed structures with a density substantially different to that of the surrounding fluid, an extension of the original methodology is required to avoid numerical instabilities. In this work, both issues will be addressed and the original framework extended to allow for multi-phase flows by means of the Level Set Method [6]. The efficient iterative solution of the corresponding non-constant diffusion (anisotropic) pressure-Poisson equation that arises in the modified fractional step method is then solved efficiently using a geometric multigrid solver in combination with segregation of the rigid constraints.

2. Methodology

In immersed techniques an underlying Eulerian mesh is employed to discretise the fluid. For the case of an incompressible Newtonian fluid, the time-dependent Navier-Stokes equations tend to be solved (due to computational efficiency) by means of a fractional step method that uncouples velocity and pressure unknowns. In order to extend the methodology first for multiphase flows, we consider the continuum domain $\Omega \subset \mathbb{R}^n$, $n = 2, 3$ and a partition into disjoint sets Ω_i that will represent each of the possible fluid phases, i.e. $\Omega = \cup_i \Omega_i$, $\Omega_i \cap \Omega_j = \emptyset, i \neq j$. The Level Set Method [7] is employed to "capture" the interfaces of the different phases as they evolve in time. A regularised Heaviside function H_i, i.e. a smooth approximation of the characteristic or indicator function χ_{Ω_i}, is evaluated for each phase Ω_i by means of the corresponding vector level set function $\boldsymbol{\phi}$, i.e. $H_i = H_i(\boldsymbol{\phi})$. Such regularisations are constructed so that the partition of unity property that the true characteristic functions satisfy also holds, i.e. $\sum_i H_i \equiv 1$. This identity allows us to consider the linear momentum conservation equation for a control volume $V \subset \Omega$ as

$$\int_V \frac{\partial}{\partial t} (\rho(\boldsymbol{H})\boldsymbol{u})\, dv + \int_{\partial V} (\rho(\boldsymbol{H})\boldsymbol{u} \otimes \boldsymbol{u} + p\boldsymbol{I} - \sigma'(\boldsymbol{H})) \cdot \boldsymbol{n} da = \int_V \boldsymbol{g} dv$$

where $H_i = [\boldsymbol{H}]_i$ is the vector of regularised Heaviside functions, $\rho(\boldsymbol{H})$ is the (non-constant, space-varying) density of the fluid as a function of the regularised Heaviside vector $\boldsymbol{H}$ and $\sigma'(\boldsymbol{H})$ is the deviatoric component of the stress tensor of the corresponding continuum phase. In addition, for an incompressible fluid, the following constraint has to be satisfied

$$\nabla \cdot \boldsymbol{u} = 0.$$

Upon solution by means of a fractional step method, the following non-constant diffusion (anisotropic) Poisson equation has to be solved to determine the pressure field

$$\nabla \cdot \left(\frac{1}{\rho(\boldsymbol{H})} \nabla \psi \right) = div(\boldsymbol{u}^*),$$

where ψ is an increment to the pressure field and $\boldsymbol{u}^*$ is an intermediate stage, non-divergence free, approximation to the velocity field. This is accomplished efficiently by use of a geometric multigrid Poisson solver. The above methodology allows for the simulation of multi-phase flows with high ratios of physical phase properties (e.g. density or viscosity).

For the inclusion of a deformable structure with initial domain Ω_0^s immersed in the fluid domain Ω, the body force $\boldsymbol{g}$ above is computed using the ISPM [1, 2], based on a Marker and Cell (MAC) spatial discretisation. A deviatoric energy functional Π^s corresponding to the deformable structure is integrated using a quadrature rule with integration points a_p and weights W^{a_p}, thus "tracking" in a Lagrangian fashion the mechanical response of the flexible structure

$$\Pi^s(\varphi) = \int_{\Omega_0^s} \hat{\Psi}^s(\varphi) dV \approx \sum_{a_p} \hat{\Psi}^s(\varphi^{a_p}) W^{a_p},$$

where φ is the solid mapping and $\hat{\Psi}^s$ is the deviatoric strain energy density. The FSI forces term $\boldsymbol{g}$ can be computed then as

$$f_i^{A_i} = \int_{\Omega_0^s} \tau_i^{'s} \cdot \nabla \zeta^{A_i}(\boldsymbol{x^s}) dV \approx \sum_{a_p} W^{a_p} \tau_i^{'s,a_p} \cdot \nabla \zeta^{A_i}(\boldsymbol{x}^{a_p}),\ g_i^{A_i} = f_i^{A_i} / (\prod_{j=1}^{j=n} \Delta x_j) \quad i = x_1, \ldots, x_n$$

where ζ^{A_i} is a suitable interpolating kernel function [2] centred at edge A_i. In the original methodology [1], the FSI interaction force $\boldsymbol{g}$ included both the deviatoric contribution to the interaction forces and the inertial term, due to the different densities of solid and fluid phases. Unfortunately, in the case of explicit time integration algorithms, the inertial contribution can lead to unstable computations for large density ratios. In the proposed extension, the ISPM is used to compute only the deviatoric component, leaving the inertial contribution to be dealt with in an Eulerian manner by the above multi-phase fluid solver.

Another limitation of the original ISPM is the modelling of very stiff or rigid structures, as the corresponding interaction term dominates the stability of the fluid solver and forces extremely small time-steps. For a rigid body occupying $\Omega^{RB} \subset \Omega$, the following additional constraint on the velocity field has to be fulfilled

$$\boldsymbol{u} = \boldsymbol{\omega} \times (\boldsymbol{x} - \boldsymbol{x_0})$$

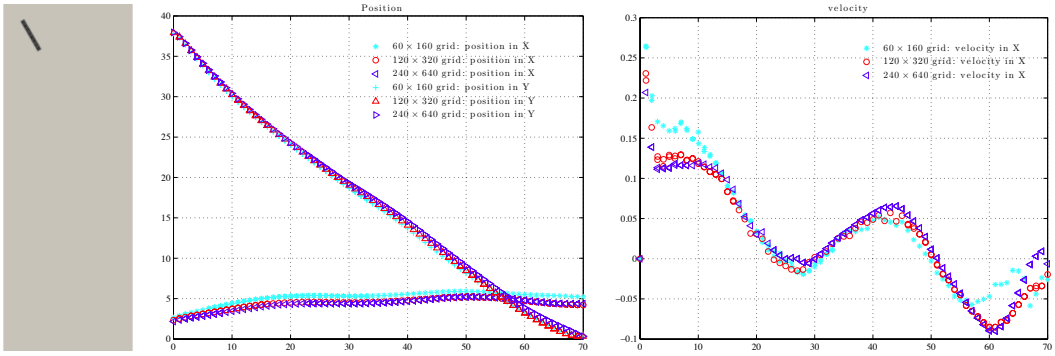
where $\boldsymbol{x_0}$ is the instant centre of rotation and $\boldsymbol{\omega}$ is the angular velocity. The rigid body is "tracked" in a Lagrangian way using a collection of integration points, in a similar fashion to the ISPM. A Lagrange multiplier field $\boldsymbol{\lambda}$ is added to the formulation to enforce the above constraint. In order to simplify the enforcement of such constraint, a weighted Least-Squares projection of the velocity field is performed. The spatial semi-discretisation is carried out using a staggered Finite Volume scheme on a Cartesian standard Marker-and-Cell (MAC) grid, where the level set and the pressure field are defined at the cell centres and the normal component of velocities, Lagrange multipliers and forces are defined at the cell faces, arriving at

$$\begin{aligned}
\frac{\mathcal{M}}{\Delta t}\boldsymbol{U}_{n+1} + \mathcal{G}\boldsymbol{P}_{n+1} + \mathcal{H}\lambda_{n+1} &= \frac{\mathcal{M}}{\Delta t}\boldsymbol{U}_n - C(\boldsymbol{U})_{n+1/2}(\rho(\boldsymbol{\phi})\boldsymbol{U}) - \mathcal{V}(\boldsymbol{U}_{n+1}) + G(\boldsymbol{U}_{n+1}),\\
\mathcal{D}\boldsymbol{U}_{n+1} &= 0,\\
\mathcal{A}\boldsymbol{U}_{n+1} &= \boldsymbol{U}_R,\\
\frac{\boldsymbol{\phi}_{n+1} - \boldsymbol{\phi}_n}{\Delta t} + C(\boldsymbol{U})_{n+1/2}(\boldsymbol{\phi}) &= 0,
\end{aligned}$$

where $\mathcal{M}, \mathcal{G}, C, \mathcal{V}, \mathcal{D}$ and $\mathcal{A}$ denote the discrete mass, gradient, convective, viscous term, divergence and Least-Squares projection operator respectively, and $\boldsymbol{U}_n$, $\boldsymbol{P}_n$ and λ_n denote the discrete velocity, pressure and Lagrange multiplier at time t_n. Note that $\mathcal{D}$ and $\mathcal{H}$ are the adjoint operators of $\mathcal{G}$ and $\mathcal{A}$ respectively. The above discrete system is solved by means of the fractional step method in conjunction with an Uzawa-type algorithm.

3. Numerical examples

In this section we present the numerical simulation of the sinking and fluttering of a rigid body in a viscous fluid with the framework presented above. The physical domain is the rectangle 15×40, discretised with a series of Cartesian meshes, the finest of which is composed of 240×640 cells, filled with a Newtonian viscous fluid of viscosity $\mu = 10^{-5}$ and density $\rho = 10^3$. A rigid rectangle of size 5×0.5 m is rotated clockwise an angle of $\pi/3$ and translated such that its geometrical centre is at position $(3.4665, 35.96)$ with respect to the bottom left corner of the fluid domain (see Figure 1a). The rigid solid has a density 1.5 times that of the fluid and is discretised using 5760 integration points. The total run-time for the case with the finest mesh is 1 hour using a 2.4 GHz Intel Core 2 Duo CPU. In Figure 1b and 1c we can observe convergence of the evolution over time of the position and velocity of the bottom left corner of the rectangle for a series of discretisations. In Figure 2, in a series of snapshots of the solution, it can be observed how the rigid solid sinks and flutters as it creates vortices in its wake.



(a) (b) (c)

Figure 1: (a) Geometry of the problem; (b) Evolution with respect to time of the x and y position of the bottom left corner of the rigid rectangle for a series of mesh discretisations; (c) Corresponding velocity of the same point with respect to time for a series of meshes.

4. Conclusions

In this work, we presented a general unified framework for the simulation of incompressible rigid/flexible/multi-phase flow problems that offers a series of advantages. First, the use of a novel

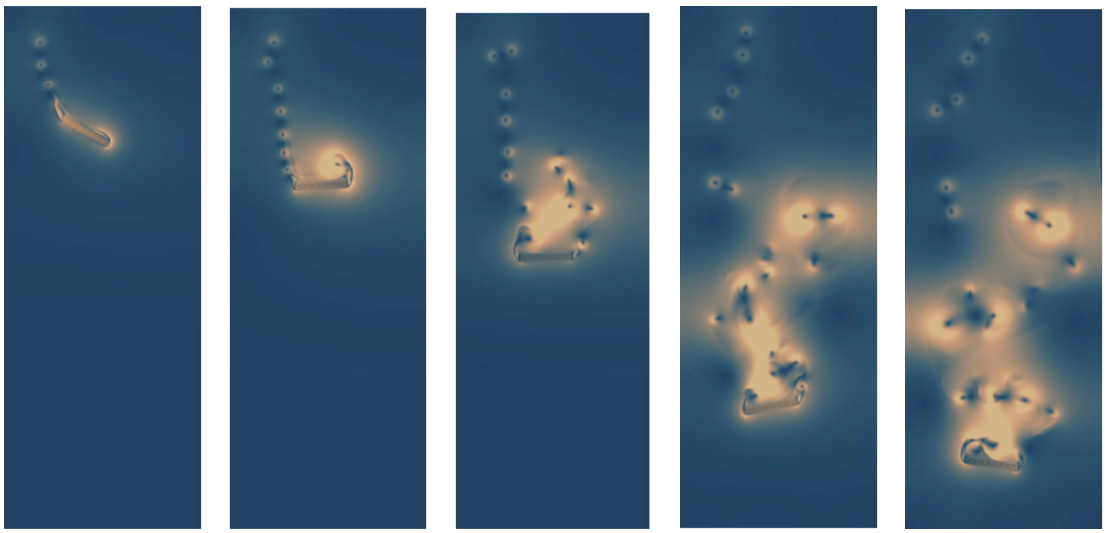


Figure 2: Snapshots of the solution for the problem of a rigid solid sinking and tumbling in a viscous fluid.

weighted least square projection scheme allows for the easy incorporation of complex rigid body motions. Second, all constraints are segregated in the system, combining fractional steps and the augmented Lagrange Method, reducing the overall computational cost. By means of an efficient geometric multigrid Poisson solver, it is shown that the framework can also be used to simulate scenarios with large density ratios between phases and/or solids.

Acknowledgements

This work is financially supported by the Zienkiewicz Scholarship, College of Engineering, Swansea University.

References

[1] A. Gil, A. Arranz Carreño, J. Bonet, O. Hassan, The immersed structural potential method for haemodynamic applications, Journal of Computational Physics 229 (2010) 8613–8641. doi:10.1016/j.jcp.2010.08.005.

[2] A. J. Gil, A. Arranz Carreño, J. Bonet, O. Hassan, An enhanced Immersed Structural Potential Method for fluid–structure interaction, Journal of Computational Physics 250 (0) (2013) 178 – 205. doi:10.1016/j.jcp.2013.05.011.

[3] C. Wood, A. J. Gil, O. Hassan, J. Bonet, A partitioned coupling approach for dynamic fluid-structure interaction with applications to biological membranes, International Journal for Numerical Methods in Fluids 57 (5) (2008) 555–581. doi:10.1002/fld.1815.

[4] C. S. Peskin, Flow patterns around heart values: a numerical method, Journal of Computational Physics 10 (1972) 252–271.

[5] K. Taira, T. Colonius, The immersed boundary method: A projection approach, Journal of Computational Physics 225 (2) (2007) 2118 – 2137. doi:10.1016/j.jcp.2007.03.005.

[6] M. Sussman, P. Smereka, S. Osher, A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow, Journal of Computational Physics 114 (1) (1994) 146 – 159. doi:10.1006/jcph.1994.1155.

[7] J. A. Sethian, Level Set Methods and Fast Marching Methods: Evolving Interfaces in Computational Geometry, Fluid Mechanics, Computer Vision, and Materials Science, 2nd Edition, Cambridge University Press, 1999.